International Journal of Instrumentation Control and Automation

Volume 2 | Issue 1 Article 7

April 2012

FINITE ELEMENT ANALYSIS OF WELDED JOINTS

RAJLAXMI N. MHETRE
Mechanical Engineering Dept. VJTI, Mumbai, India, rajlaxmi.mhetre@gmail.com

S.G. JADHAV
Mechanical Engineering Dept. VJTI, Mumbai, India, sgjadhav@vjti.org.in

Recommended Citation
MHETRE, RAJLAXMI N. and JADHAV, S.G. (2012) "FINITE ELEMENT ANALYSIS OF WELDED JOINTS," *International Journal of Instrumentation Control and Automation*: Vol. 2 : Iss. 1 , Article 7.
DOI: 10.47893/IJICA.2012.1064
Available at: https://www.interscience.in/ijica/vol2/iss1/7

FINITE ELEMENT ANALYSIS OF WELDED JOINTS

RAJLAXMI N. MHETRE[1] & S.G. JADHAV[2]

[1&2]Mechanical Engineering Dept. VJTI, Mumbai, India
Email : rajlaxmi.mhetre@gmail.com, sgjadhav@vjti.org.in

Abstract— Fusion welding is one of the most used methods for joining metals. This method has largely been developed by experiments, i.e. trial and error. The problem of distortion and residual stresses of a structure is due to welding is important to control. Industry where the components are expensive and safety and quality are highly important issues. The aim of the work presented in this paper is to develop an efficient and reliable method for simulation of the welding process using the Finite Element Method. The method may then be used when designing and planning the manufacturer of a component, so that introduction of new components can be made with as little disturbance as possible. When creating a numerical model, the aim is to implement the physical behavior of the process into the model. However, it may be necessary to compromise between accuracy of the model and the required computational time. Different types of simplifications of the problem and more efficient computation methods are discussed. Simulations have been carried out in order to validate the models. Moving heat source and element death & birth technique is used to simulate Welding Process.

Keywords*- Finite Element Analysis; Moving Heat Source; Mechanical Analysis.*

I. INTRODUCTION

Material joining is one of the major manufacturing processes used to assemble metallic and non-metallic parts for several applications. The industry is actively considering a number of alternate welding technologies that would enable the increased use of lightweight and high performance materials. In the process of welding, the high temperature differences result in large temperature strains, which affect the distribution of the contact pressure between structural components.

The safety requirements and the high costs of performing experiments to find different manufacturing routes is the motivation to increase the use of simulations in design of components as well as its manufacturing. It is then possible to optimize a chain of manufacturing processes as, for example, the welding residual stresses will affect the deformations during a subsequent heat treatment. The aim of the work presented in this thesis is to develop an efficient and reliable method for simulation of the welding process using the Finite Element Method.

The method may then be used when designing and planning the manufacturer of a component, so that introduction of new components can be made with as little disturbance as possible. In the same time the developed tool will be suitable for the task to perform an optimal design for manufacturing. Whilst this development will also be valuable in predicting the components subsequent in-service behavior, the key target is to ensure that designs are created which are readily manufactured. If this understanding is captured and made available to designers, true design for manufacturer will result. This will lead to right first time product introduction and minimal ongoing manufacturing costs as process capability will be understood and designed into the component.

When creating a numerical model, the aim is to implement the physical behavior of the process into the model. However, it may be necessary to compromise between accuracy of the model and the required computational time. Different types of simplifications of the problem and more efficient computation methods are discussed. Simulations have been carried out in order to validate the models.

II. FINITE ELEMENT ANALYSIS

The finite element method has become a powerful tool for the numerical solution of a wide range of engineering problems. Application range from deformation and stress analysis of automotive, aircraft, building, and bridge structures to field analysis of heat flux, fluid flow, magnetic flux, seepage, and other flow problems.

The complex nature of the welding process causes difficulty in analyzing and modelling by numerical methods. These complexities include: material and thermal properties which vary with temperature, transient heat transfer with complicated boundary conditions, moving heat sources, phase changes and transformations, complex residual stress states and the difficulties of making experimental measurements at high temperatures [1]. In addition to these complexities, finite element modelling of the weld process must include complex thermo-mechanical interactions, metallurgical transformations, material deposit, and moving heat sources. Accurate prediction of the thermal cycle in the weld joint is the first step when predicting residual stresses. Prediction of the temperature field requires a non-linear, 3-D analysis with temperature dependent material properties. Moving heat sources are used to generate the temperature fields during the welding process and material deposit is implemented using a variety of means. The temperature history is used for the calculation of the thermal stresses and displacement fields during and after the welding process [2]. In other words, once the thermal history is known, the plastic deformation due to the thermal expansion and contraction must be calculated. The work has been focussing on validation of the model on smaller test pieces so far. The welding method that has been studied is Gas Metal Arc Welding and the material used are IS 1570 alloy steel.

A. *Model of Welding Heat Source*

Model of Gaussian heat source is adopted for the heat distribution which is fit for actual state, and would lead to the results more accurate [8]. The heat flux density of Gaussian model is of the following form:

$$q^* = q_{max} e^{-cr^2} \qquad (1)$$

Where q^* is surface heat flux density which locates in Gaussian heat source radius r; q_{max} is maximal heat flux density in the centre of heating spot, W/m^2 c is concentrative coefficient of heat source which is relative to the welding method; r is distance from the centre of heat source, m.

B. *Equation of Welding Temperature Field*

The equation of heat process is of the following form:

$$C_p \frac{dT}{dt} - \nabla[\lambda \nabla T] + \bar{Q} \qquad (2)$$

Where C_p is specific heat in definite pressure, J/ (kg·K); λ is thermal conductivity, W/ (m·K); Q is intensity of internal heat source, W/m^3; T is temperature, K; t is time, s. The equation of boundary heat flux density caused by convection heat exchange is described as:

$$q_c = \alpha_c (T - T_{0c}) \qquad (3)$$

The equation of boundary heat flux density caused by radiant heat exchange is written as:

$$q_r = \sigma_0 \varepsilon_0 (T^4 - T_{0r}{}^4) \qquad (4)$$

Where ε_0 is emissivity; σ_0 is radiant constant, W/ (m^2·K^6); T_{0r} is the reference temperature when there is no radiation.

C. *Equation of Welding Residual Stress Field*

The mechanical equation is given by:

$$\sigma_{ij} = 0 \qquad (5)$$

Where бij is stress item which contains heat stress.

Equation of heat strain is expressed as:

$$\varepsilon_{ij}^T = \alpha_{ij}(T - T_0)\delta_{ij} \qquad (6)$$

Where $\square_{ij}^T$ is thermal strain tensor; α_{ij} is heat expand coefficient; T_0 is reference temperature; δ_{ij} is δ arithmetic operator.

Constitutive equation of stress and strain is satisfied by the following function:

$$d\sigma_{ij} = D_{ij}\left(d\varepsilon_{kl} - d^p_{\varepsilon_{kl}} - d^c_{\varepsilon_{kl}} - d^T_{\varepsilon_{kl}} \right) \qquad (7)$$

Where D_{ij} is coefficient of elastic constitutive tensor; $d\square_{kl}$, $d^p{}_{\square kl}$, $d^c{}_{\square kl}$, $d^T{}_{\square kl}$ are total strain, plastic strain, creep strain and heat strain respectively.

III. MOVING HEAT SOURCE

If material is added to or removed from a system, certain elements in your model may become “existent” or “nonexistent” [4]. The element birth and death options can be used to deactivate or reactivate selected elements in such cases. This feature can be useful in analyzing excavation as in mining and tunnelling, staged construction as in shored bridge erection, sequential assembly as in fabrication of layered computer chips, and many other applications in which element can easily be identified and activated or deactivated by their known locations.

A. Element Birth & Death Technique

To achieve the "element death" effect, the ANSYS program does not actually remove "killed" elements. Instead, it deactivates them by multiplying their stiffness or conductivity, or other analogous quantity by a severe reduction factor. This factor is set to 1.0E-6 by default, but can be given other values. Element loads associated with deactivated elements are zeroed out of the load vector; however, they still appear in element-load lists. Similarly, mass, damping, specific heat, and other such effects are set to zero for deactivated elements. The mass and energy of deactivated elements are not included in the summations over the model. An element's strain is also set to zero as soon as that element is killed. In like manner, when elements are "born," they are not actually added to the model; they are simply reactivated. all elements must be created, including those to be born in later stages of analysis, while in. New elements cannot be created in SOLUTION. To "add" an element, first deactivate it, and then reactivate it at the proper load step.

When an element is reactivated, its stiffness, mass, element loads, etc. return to their full original values. Elements are reactivated having no record of strain history or heat storage, etc. However, initial strain defined as a real constant will not be affected by birth and death operations. Also, unless large-deformation effects are turned on some element types will be reactivated in their originally specified geometric configuration (large-deformation effects should be included to obtain meaningful results). Thermal strains are computed for newly-activated elements based on the current load step temperature and the reference temperature. Thus, newborn elements with thermal loads may not be stress-free as intended.

To explain the element birth and death technique, a sample plate is considered. The size of the plate is 1065 mm X 200mm. The thickness of the plate is 12 mm. Fig.1 represents the meshing of the plate.

The heat source is added at the bottom left corner of the plate. The direction of the moving heat source is from left to right. Then temperature load is applied to the first node and the load step file is saved. When applying the temperature load to the next node the previous load is deleted. In this way after applying and removing temperature loads solve all the load step files. Moving heat source is shown in Fig.2(a) and (b). The blue colour represents the atmosphere temperature which is given in the analysis.

The weld seam undergoes complex temperature changes during welding process, which causes transient thermal stress and incompatible plastic strain in weld seam and nearby regions. The change of strain during welding process is shown as follows:

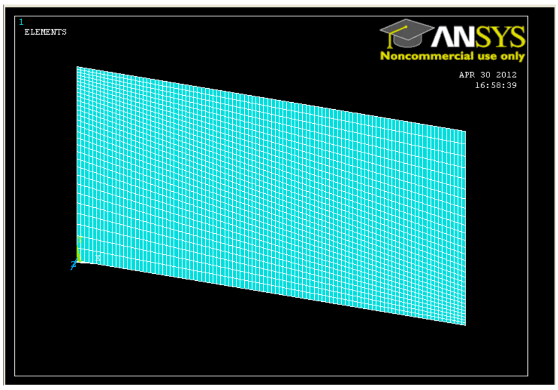

Figure 1.Sample Plate Meshing

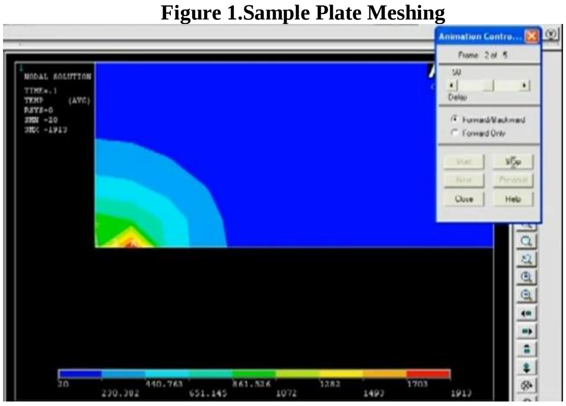

Figure 2.(a) Moving Heat Source

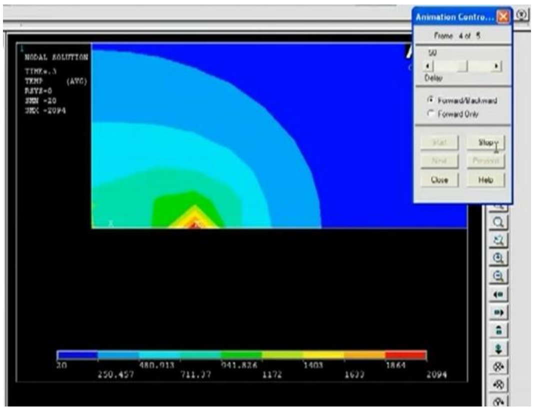

Figure 2.(b) Moving Heat Source

$$Єt = Єe + (-Єp) + (-Єel) \tag{8}$$

The transient temperature field contour of welding process when welding arc moves to the centre of sheets. It can be observed that the temperature field contour is denser in the front of molten pool and gradually becomes coarser behind away from molten pool.

IV. MECHANICAL ANALYSIS

Mechanical analysis is done using Solid works software. The analysis of sample stiffener plate is carried out. The model of the stiffener plate with weld bead is shown in fig.3. The meshing of the stiffener is shown in fig.4. The mesh used for this model is solid mesh. The Mesher is standard mesh. The 4 point Jacobian was used to generate this mesh.

Jacobian point is a scale factor arising because of transformation of co-ordinate system.

A. Loads and Restraints for StiffenerPlate

The base of the stiffener is kept fixed and the force will be acting on the upper surface of the stiffener. The side walls of the base plate are also kept fixed to carry out the simulation. The loads and constraints on stiffener plate is shown in fig.5.

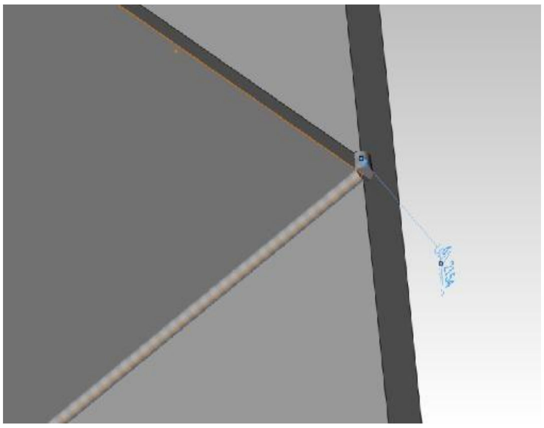

Figure 3. Stiffener Plate Model

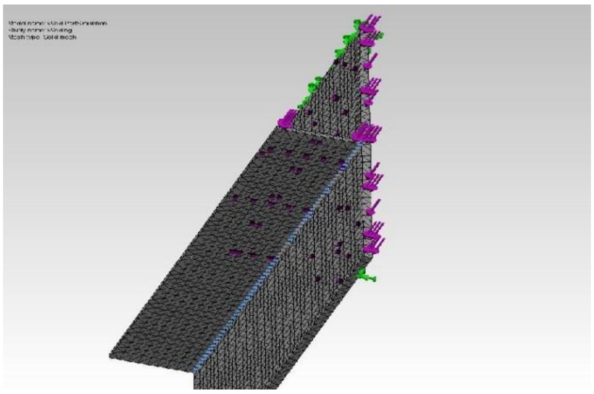

Figure 4. Stiffener Plate Meshing

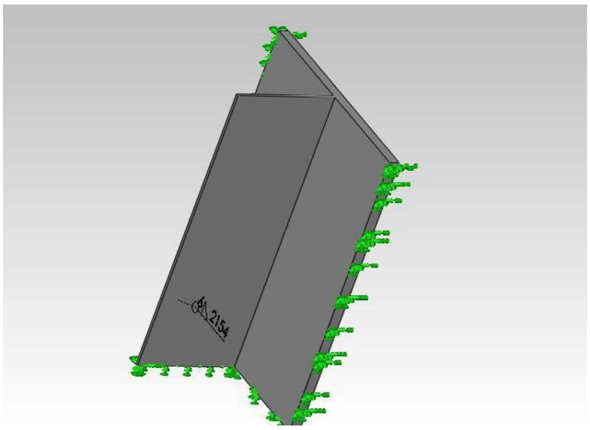

Figure 5. Loads & Constraints on Stiffener Plate

B. *Mechanical Material Properties*

The mechanical properties of the IS 1570 steel material used for the mechanical analysis are shown in table 1.

Table 1. Mechanical Material Properties

Material	**Model Type**	**Yield Strength**	**Elastic Modulus**	**Poisson's Ratio**
IS 1570 40CNi2Cr1Mo28	Linear Elastic Isotropic	415 MPa	205 GPa	0.28

The throat thickness comes out to be 6 mm for the 20 mm plate thickness.

V. RESULTS

The stress, strain and displacement analysis is carried out for the stiffener. The Stress plot is shown in fig.6. From the s analysis results it can be seen that the stresses developed are much below than the allowable stress for selected material.

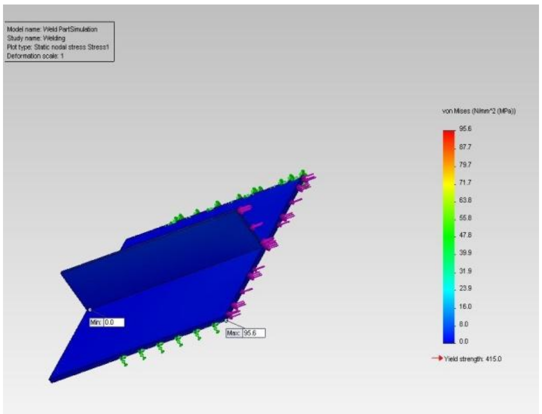

Figure 6. Stress Plot for Stiffener Plate

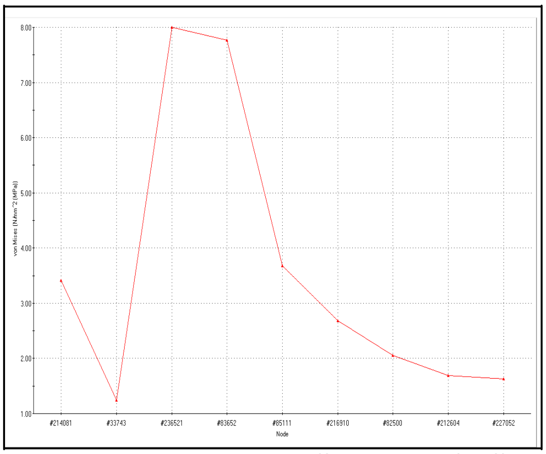

Figure 7. Stress Distribution on Different Nodes of Stiffener

The Displacement plot is shown in fig.8. The probe results along the weld path are also shown in Fig.9. The maximum displacement is on upper side of the stiffener due to the applied load.

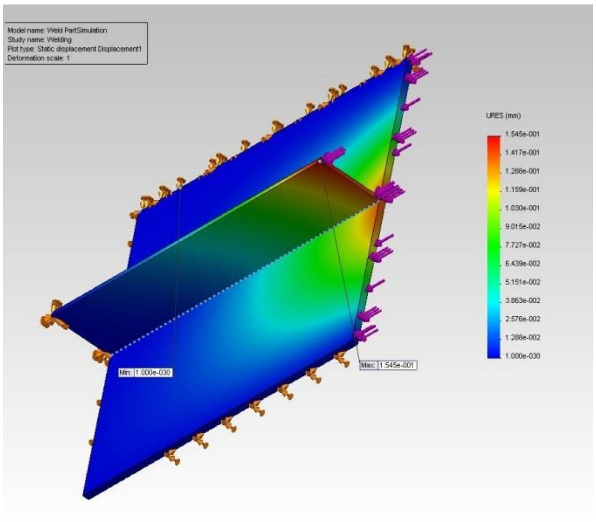

Figure 8. Displacement Plot for Stiffener

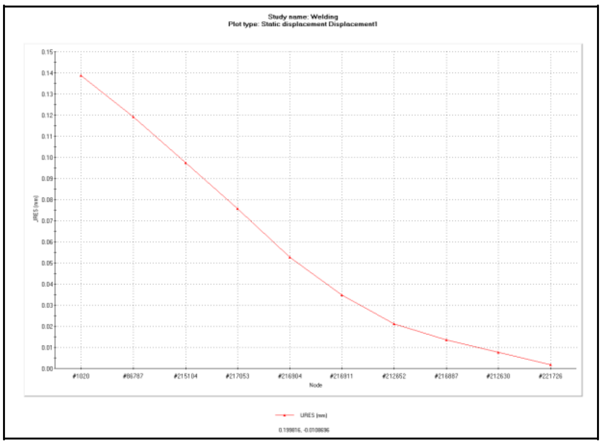

Figure 9. Displacement along the weld at different nodes

In Fig.9, the displacement along the weld at different nodes is shown. It can be seen that the maximum displacement is at the starting of the weld path. The displacement decreases towards the fixed end i.e. the base of the stiffener plate.

VI. CONCLUSION

The mathematical model of a technological process of welding has been presented. The molten state of the weld pool is presented by the moving heat source. The element birth and death technique was introduced to represent the melting of the material.

A numerical simulation of the welding induced temperature field and corresponding distortions helps to investigate the heat effects of welding and, in the long run, to optimize the quality of welded parts. Due to the complexity of the welding process itself, the material property data for the simulation has to be temperature and phase dependent.

The analysis results help us understand the phenomena governing the welding of a joint, offering insight on the mechanisms and mechanical aspect particular to the welding process. Having understood the welding mechanism, the effects of the welding can be better quantified and therefore can be better addressed in the early stages of the design.

ACKNOWLEDGMENT

The authors wish to thank Esmech Equipment Pvt.Ltd. and VJTI, Mumbai for sponsoring this project. The project work was carried out in Esmech Equipment Pvt. Ltd. under Memorandum of Understanding with VJTI, Mumbai. Resources were provided by EEPL for the completion of this project work.

REFERENCES

[1] Aravinthan and C. Nachimani ,2011 ” Analysis of Spot Weld Growth on Mild and Stainless Steel “,Supplement To The Welding Journal, August 2011 Sponsored by the American Welding Society and the Welding Research Council.

[2] Daniel Gutscher, Transportation Technology Center, Inc**.,** 2009, AAR Rail Welding Research – Recent Developments.

[3] Guodong Zhang, Changyu Zhou, 2007,” Numerical simulation of creep for welded joint under consideration of welding residual stress”, Journal of Pressure Equipment and Systems 5 (2007) 24-28.

[4] M. De Strycker, P. Lava, W. Van Paepegem, L. Schueremans, And D. Debruyne,June 2011,” Measuring Welding Deformations with the Digital Image Correlation Technique”, SUPPLEMENT TO THE WELDING JOURNAL, JUNE 2011 Sponsored by the American Welding Society and the Welding Research Council.

[5] M. Baboi And D. Grewell,” Comparison of Control Algorithms for Ultrasonic Welding of Aluminium “, welding journal.

[6] C. Schwenk And M. Rethmeier, Nov. 2011 Vol.90,” Material Properties for Welding Simulation —Measurement, Analysis, and Exemplary Data”, Welding Journal.

[7] D. Radaj, ‘Heat Effects of Welding – Temperature field residual stress distortion’, Springer – Verlag, Berlin, 1992.

[8] Dong ZB, Wei YH, Liu RP. Three dimension simulation of thermal distributions of welding stainless steels. Transactions of the China welding institution 2004; 25(2): 9-14.

[9] Oddy, A. S., Goldak, J.A., and McDill, J. M. J., 1989, "Transformation Plasticity and Residual Stresses in Single- Pass Repair Welds," ASME *PVP- Weld Residual Stresses and Plastic Deformation*, **173**, pp. 13-18.

[10] Tsai, C. L., Park, S. C., and Cheng, W. T., 1999, "Welding Distortion of a Thin-Plate Panel Structure," *Welding Journal, Welding Research Supplement*, pp. 156s - 165s.

◆ ◆ ◆